STRATEGIES FOR COUPLING ENERGY SIMULATION AND COMPUTATIONAL FLUID DYNAMICS PROGRAMS

Zhiqiang Zhai and Qingyan Chen
Massachusetts Institute of Technology
Cambridge, MA 02139, USA

Joseph H. Klems and Philip Haves
Lawrence Berkeley National Laboratory
Berkeley, CA 94720, USA

ABSTRACT

Energy simulation (ES) and computational fluid dynamics (CFD) can play important roles in building design by providing complementary information about the buildings' environmental performance. However, separate applications of ES and CFD are usually unable to give an accurate prediction of building performance due to the assumptions involved in the separate calculations. Integration of ES and CFD eliminates many of these assumptions since the information provided by the models is complementary. Several different approaches to integrating ES and CFD are described. In order to bridge the discontinuities of time-scale, spatial resolution and computing speed between ES and CFD programs, a staged coupling strategy for different problems is proposed. The paper illustrates a typical dynamic coupling process by means of an example implemented using the EnergyPlus and MIT-CFD programs.

Key words: energy simulation, computational fluid dynamics (CFD), integration, building design

INTRODUCTION

Energy simulation and computational fluid dynamics programs provide complementary information about the performance of buildings. ES programs, such as EnergyPlus (Crawley et al 2000), address the performance of the building envelope, as well as the heating, ventilating and air conditioning (HVAC) system, and provide the whole building energy analysis. Space-averaged indoor environmental conditions, cooling/heating loads, coil loads, and energy consumption can be obtained on an hourly or sub-hourly basis for periods of time ranging from a design day to a reference year or more. CFD programs, on the other hand, make detailed predictions of thermal comfort and indoor air quality (IAQ), including the distributions of air velocity, temperature, relative humidity and contaminant concentrations. The distributions can be used further to determine indices such as the predicted mean vote (PMV), the percentage of people dissatisfied (PPD) due to discomfort, the percentage dissatisfied (PD) due to draft, and ventilation effectiveness. With the information from both ES and CFD calculations, designers can design environmental control systems for buildings that satisfy multiple criteria.

However, due to the complete mixing model used in ES, most ES programs cannot accurately predict energy for systems that produce non-uniform air temperature distributions in the occupied space, such as displacement ventilation systems. Moreover, the spatially averaged comfort information generated by the single node model of ES cannot satisfy advanced design requirements. The convective heat transfer coefficients used in ES programs are usually empirical and may not have general applicability, either. Furthermore, most ES programs are unable to provide information on the airflow entering a building, for example, by natural ventilation, while the ventilation rate information is very important for predicting room air temperature and (or) heating/ cooling load.

CFD, on the other hand, can easily determine the temperature distribution and convective heat transfer coefficients, which ES needs. CFD is also a powerful tool for the simulation of natural ventilation driven by wind effect, stack effect, or both. At the same time, CFD also needs information from ES as inputs, such as air conditioning loads and surface temperatures. Otherwise, CFD has to compute results based on estimated boundary conditions.

Therefore, coupling ES with CFD is very attractive and is the objective of the present investigation. Starting from the principles of ES and CFD, the paper describes possible approaches to ES and CFD coupling. The current study emphasizes the explicit coupling of individual ES and CFD programs by exchanging the inter-coupled boundary values.

In order to bridge the disparities between ES and CFD programs due to the different physical models and numerical methods employed, the study suggests the staged coupling processes that may reduce the computing demands while keeping the advantages of coupled calculations. To demonstrate the process and benefits of coupled simulation, examples of coupled calculations for a simple office space are presented at the end of this paper.

FUNDAMENTALS OF ES AND CFD THERMAL COUPLING

Principle of ES

Energy balance equations for zone air and surface heat transfer are two essential equations that an energy program should solve. The energy balance equation for room air is

$$\sum_{i=1}^{N} q_{i,c}A_i + Q_{other} - Q_{heat\_extraction} = \frac{\rho V_{room} C_p \Delta T}{\Delta t} \qquad (1)$$

where

$\sum_{i=1}^{N} q_{i,c}A_i$ = convective heat transfer from enclosure surfaces to room air
$q_{i,c}$ = convective flux from surface i
N = number of enclosure surfaces
A_i = area of surface i
Q_{other} = heat gains from lights, people, appliances, infiltration, etc.
$Q_{heat\_extraction}$ = heat extraction rate of the room
$\frac{\rho V_{room} C_p \Delta T}{\Delta t}$ = room air energy change
ρ = air density
V_{room} = room volume
C_p = air specific heat
ΔT = temperature change of room air
Δt = sampling time interval, normally one hour

The heat extraction rate is the same as the cooling/heating load when the room air temperature is maintained constant ($\Delta T = 0$). The convective heat fluxes are determined from the energy balance equations for the corresponding surfaces, as shown in Figure 1. A similar energy balance is performed for each window. The surface energy balance equation can be written as:

$$q_i + q_{ir} = \sum_{k=1}^{N} q_{ik} + q_{i,c} \qquad (2)$$

where

q_i = conductive heat flux on surface i
q_{ir} = radiative heat flux from internal heat sources and solar radiation
q_{ik} = radiative heat flux from surface i to surface k

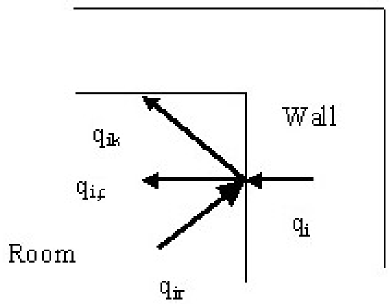

Figure 1. Energy balance on the interior surface of a wall, ceiling, floor, roof or slab

The q_i can be determined by transfer functions, weighting factors, or by solutions of the discretized heat conduction equation for the wall using finite differences. The radiative heat flux is

$$q_{ik} = h_{ik,r}(T_i - T_k) \qquad (3)$$

where

$h_{ik,r}$ = linearized radiative heat transfer coefficient between surfaces i and k
T_i = temperature of interior surface i
T_k = temperature of interior surface k

And

$$q_{i,c} = h_c (T_i - T_{room}) \qquad (4)$$

where

h_c = convective heat transfer coefficient
T_{room} = room air temperature

The convective heat transfer coefficient, h_c, is unknown. Most energy programs estimate h_c by empirical equations or as a constant. If the room air temperature, T_{room}, is assumed to be uniform and known, the interior surface temperatures, T_i, can be determined by simultaneous solving the surface heat balance equations (2).

Space cooling/heating load then can be determined from the calculated convective heat transfer from enclosure surfaces using Equation (1). Thereafter, the coil load is determined from the heat extraction rate and the corresponding air handling processes and HVAC system selected. With a plant model and hour-by-hour calculation of the coil load, the energy consumption of the HVAC system for a building can be determined.

Principle of CFD

CFD is the application of numerical techniques to solve the Navier-Stokes (N-S) equations for fluid flow. The N-S equations are derived by applying the principles of conservation of mass and momentum to a control volume of fluid (A thorough treatment may be found in many textbooks on CFD). When applying CFD to the IAQ and thermal comfort problem, the conservation of mass for a contaminant species and energy for thermal responses also may be applied. All of the conservative governing equations may be written in the following general form:

$$\frac{\partial \Phi}{\partial t} + (V \bullet \nabla)\Phi - \Gamma_\phi \nabla^2 \Phi = S_\phi \qquad (5)$$

where

t = time

Φ = V_j for the air velocity component in the j direction
= 1 for mass continuity
= T for temperature
= C for different gas species
= turbulence parameters

V = velocity vector

Γ_ϕ = diffusion coefficient

S_ϕ = source term

Multiple concentrations, C, can be used to simulate different species, such as water vapor and various contaminants. For buoyancy-driven flows, the Buossinesq approximation, which ignores the effect of pressure changes on density, is usually employed. The buoyancy-driven force is treated as a source term in the momentum equations. Because most practical flows are turbulent, a turbulence model must be applied for most indoor airflow in order to make the flow solvable with present computer capacity and memory.

The flow governing equations are highly non-linear and self-coupled, which make it impossible to obtain analytically exact solutions for most room air flows. Therefore, in CFD, the equations are solved by discretizing the equations using the finite volume method that converts them to a set of numerically solvable algebraic equations. The spatial continuum is divided into a finite number of discrete cells, and finite time-steps are used for dynamic problems. After generating a reasonably fine numerical grid on which the discrete algebraic equations will be solved, and specifying a set of problem-dependent boundary conditions, the calculation can be iterated automatically until a prescribed convergence criterion is met.

As with all modeling techniques, the accuracy of CFD prediction is highly sensitive to the boundary conditions supplied (assumed) by the user. Essentially, the flow inside the CFD solution domain (i.e., a room) is driven by the boundary conditions. Normally, the boundary conditions for CFD simulation of indoor airflows relate to the inlet (supply), outlet (exhaust), enclosure surfaces, and internal objects. The temperature, velocity and turbulence of the air entering from diffusers or windows determine the inlet conditions, while the interior surface temperatures and/or heat fluxes are important thermal boundary conditions for the enclosures.

Coupling Approaches

The above discussion of the principles of ES and CFD shows that the convective heat transfer from interior surfaces of a space not only links the zone air energy balance equation with the enclosure energy balance equation in ES, but also links ES with CFD. The problem of model coupling is, then, focused on how to treat the convective heat transfer in ES and CFD.

Depending on the method used to treat the convective heat transfer, two different coupling approaches are possible in practice. Since CFD solves the energy equation for the indoor air, a CFD program can be extended to solve heat transfer in solid materials, such as building enclosures, with an appropriate radiation model. The convective heat transfer is then calculated directly in the simulation. This is the conjugate heat transfer method. Some researchers have applied this method to integrated calculations (e.g. Holmes et al 1990, Chen et al 1995, Moser et al 1995, Schild 1997). This approach is powerful in predicting all the information from one calculation but is very expensive computationally (Chen et al 1995). The reason for this is twofold. First, when the CFD calculates the heat transfer in solid materials, the calculation becomes stiffer and the computing time goes up dramatically (Thompson and Leaf 1988). In order to reach a consistent solution between the characteristic time of air (a few seconds) and the thermal response time of the building envelope (a few hours), extended CFD simulations must be performed over a period comparable to the thermal response time of the building envelope, but with a time step as small as the characteristic thermal time of the air (a few seconds). It is then necessary to repeat the computationally demanding calculation many times. The second part of the reason lies in the exponential way that CFD calculations grow with building size. Hence, the conjugate heat transfer method is not practical for immediate use in a design context with current computer capabilities.

The alternative approach is to couple ES and CFD programs directly and exchange the convective heat transfer information between the two programs. In principle, a fully iterated ES and CFD coupling program can provide a solution that is equivalent to the conjugate heat transfer method, provided that the

ES program subdivides surfaces sufficiently to model any important temperature variations. However, since the CFD program coupled with ES is not involved in the transient calculation, the CFD solution at a specific time step is actually quasi-steady, consistent with the given boundary conditions for that time step. Such a calculation, thus, has the advantage that it does not attempt to solve the flow field during the transition from one time step to the next, and therefore greatly saves on computing time. Chen (1988) coupled an energy simulation program with a CFD program and demonstrated the importance of temperature stratifications in the determination of air conditioning loads. Srebric (1999) improved Chen's study with manual run-time coupling calculations for several typical cases. The ESP-r program (Negrao 1995, Clarke et al 1995a, 1995b, Beausoleil-Morrison 2000) has also integrated a CFD solver (dfs) into a whole-building simulation environment, using three handshaking methods. Most code coupling practices identified so far have indicated that the code coupling approach is able to obtain reasonable solutions with acceptable computing efforts. Since the code coupling approach is superior to the first one in terms of its practical applicability, this paper focuses on the discussion of this approach.

Coupling Principle of ES and CFD Codes

In the code coupling approach, convective heat transfer from enclosures is the most important information for coupling. It is crucial for the accurate calculation of the cooling/heating load and energy analysis in ES, as well as for the accurate specification of boundary conditions in CFD. The air temperature in the boundary layer of a surface and the convective heat transfer coefficient are two key factors determining the convective heat transfer. However, most ES programs adopt the complete mixing air model in solving the energy balance equation for room air. That is, the programs assume that the room air temperature is uniform, although this is often not the case. Actually, it is easy to use CFD to determine the air temperatures near the surfaces from the air temperature distribution. In addition, instead of using empirical equations, the convective heat transfer coefficients can also be determined from the CFD simulation:

$$h_{i,c} = C_p \frac{\mu_{eff}}{Pr} \frac{1}{\Delta x} \qquad (6)$$

where

C_p = air specific heat

μ_{eff} = effective kinetic viscosity, heavily dependent on turbulence models

Pr = Prandtl number

Δx = normal distance from a point near a wall to the wall

A straightforward coupling method is to pass the surface average of the air temperature $T_{i,air}$ computed by CFD at the flow grid next to the wall surface, together with the corresponding averaged convective heat transfer coefficient, $h_{i,c}$, to ES.

The $T_{i,air}$ and $h_{i,c}$ provided by CFD should be used in ES by modifying Equation (4) to:

$$q_{i,c} = h_{i,c} (T_i - T_{i,air}) = h_{i,c}(T_i - T_{room}) - h_{i,c} \Delta T_{i,air} \qquad (7)$$

where $\Delta T_{i,air} = T_{i,air} - T_{room}$ and T_{room} is the design air temperature in the room. After finishing each CFD simulation, ES obtains the updated $T_{i,air}$ and $h_{i,c}$ from CFD, substituting them into Equation (7). Then, ES can solve heat balance equations (2) and (1) sequentially with this new Equation (7) to obtain new surface temperatures, room air temperature and air-conditioning load.

CFD computation, on the other hand, needs the interior surface temperatures and/or heat fluxes as the boundary conditions for the space. These values are the direct results of the energy calculation. In addition, the heat extraction rate from ES is also needed to determine the inlet boundary conditions in CFD calculation.

$$Q_{heat\_extraction} = \rho C_p A V (T_{supply} - T_{outlet}) \qquad (8)$$

where

C_p = air specific heat

A = diffuser air supply area

V = supply air velocity

T_{supply} = supply air temperature

T_{outlet} = return air temperature

For a constant air volume (CAV) HVAC system, T_{supply} is variable depending on $Q_{heat\_extraction}$, while V is a constant. For a variable air volume (VAV) system, T_{supply} is constant, while V is a variable.

By exchanging this complementary information, ES and CFD become tightly coupled. Since, in the building, the heat flows and surface temperatures vary with time, in theory, it is necessary to do a CFD calculation for each time step. Even at each time step, iteration may be needed to reach mutually consistent results between ES and CFD. The structure of the coupled simulation is illustrated in Figure 2.

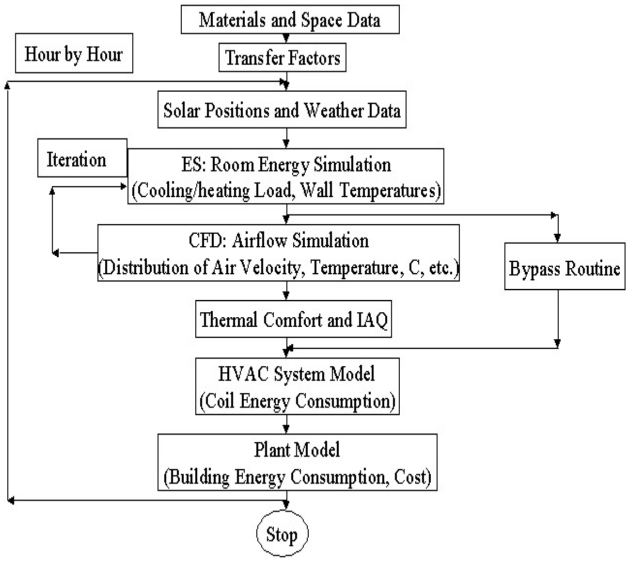

Figure 2. Structure of coupling simulation

STAGED STRATEGY FOR ES/CFD COUPLING

Although the idea and principle of the code coupling approach is straightforward, the coupling is challenging in practice due to the considerable disparities of the physical models and numerical schemes between ES and CFD programs. Three main discontinuities exist between ES and CFD programs. The first one is a time-scale discontinuity: ES has a characteristic time-scale of hours for building performance, but CFD is on the order of a few seconds for air. The second is a modeling discontinuity: the indoor environmental conditions predicted for each space in ES are a spatial average, while CFD presents field distributions of the variables. The last one is a speed discontinuity: the execution time for energy simulation is of the order of a few seconds per zone per year, and the memory requirement is small (about 1 Mb), while a three-dimensional CFD calculation for a zone may take a few hours to a few days and require about 100 Mb of memory, even for a modest grid size (Srebric et al 1999).

To bridge these discontinuities between ES and CFD, special coupling strategies need to be developed. For the time-scale discontinuity, the current coupling strategy actually partitions the whole calculation into a long-time-scale process in ES, such as the HVAC system, internal loads, varying weather, time schedule, and the heat flows in the building structure, and a short-time-scale process (strictly speaking, a quasi-static process at a given time-step) modeled by CFD. Space model discontinuity can also be bridged by appropriate numerical approximation, although the effects of different numerical approximation algorithms on the coupling performance need to be investigated further, both in theory and in practice. However, the computational demands of CFD simulation make the coupling almost impractical for most real situations. In addition to using more numerical approximations, such as simpler turbulence models, to reduce the computing time of CFD programs directly, it is necessary to develop special coupling strategies to minimize the number of CFD calculations necessary. The present study proposes a staged coupling strategy that is closely related to the physics of the problems to be studied. The staged couplings are called *static coupling* and *dynamic coupling*, respectively, as listed and illustrated in Table 1. The definitions of "static" and "dynamic" coupling come from the operating behaviors of coupling; that is, the dynamic coupling process performs continuous (dynamic) information exchange while the static coupling process has occasional (static) information exchange for the whole simulation period.

Static coupling involves one-step or two-step exchange of information between ES and CFD programs, depending on the building performance and resolution requirement. Due to the few coupling steps and the static feature of the coupling operation, static coupling can usually be performed manually, with few changes in ES and CFD codes. Generally, the one-step static coupling is good in the cases where ES or CFD or both are not very sensitive to the exchanged variables. For example, ES is rather insensitive to $\Delta T_{i,air}$ and $h_{i,c,}$ in an air-conditioned room with low velocity mixing ventilation, while CFD needs inlet conditions and wall temperatures as inputs, so the one-step static coupling from ES to CFD is a good choice. If the information from CFD, such as $h_{i,c}$, differs significantly from that used in the first ES calculation, ES may take this information from CFD for the new energy and temperature calculation. This is the ES-CFD-ES two-step static coupling. Two-step static coupling is good enough for buildings in which the changes in the exchanged information are not significant, and the solution is not strongly dependent on the exchanged data.

Dynamic coupling, which involves coupling between the two programs at every time step, is needed when both ES and CFD solutions depend on boundary conditions that vary significantly with time. There are four kinds of dynamic coupling. The first one is called *one-time-step dynamic coupling*, which focuses on the ES/CFD coupling at one specific time step. At this time step, the iteration between ES and CFD is performed until a mutually consistent solution is found. For the cases with significant weather or load turning points during the concerned period and with the close relationship between the solution and the exchanged data, the full iteration at these significant time-steps is necessary. More building cases involve the interest of airflow, thermal, and energy for the entire period of time. In this case, the ES/CFD coupling needs to be conducted at every time step for this period. In fact, it is not necessary to couple the two programs at every time-step if the changes of the required information are not significant. At each coupling time step, ES and CFD may iterate only one time and then move on to the next time step, which is called *quasi-dynamic coupling*. If ES and CFD iterate for a couple of times at each coupling time step or even until some convergence criteria are met and then go on to the next time step, that is the *full dynamic coupling*. Full dynamic coupling is undoubtedly the most accurate, but also most intensive computationally. One way to reduce the computational load is to use *virtual dynamic coupling.* The room air temperatures and the convective heat transfer coefficients required by ES are generated by CFD as the functions of cooling/heating loads (for conditioned spaces) or indoor outdoor air temperature difference (for unconditioned spaces). At each coupling time step, ES determines the values of $\Delta T_{i,air}$ and $h_{i,c}$ by interpolating the CFD results. Virtual dynamic coupling is suitable for buildings without dramatic changes of heat/cooling load and outdoor air temperature because the dramatic changes make the curve-fitted functions less accurate.

Table 1. Illustration of staged coupling strategy

(The arrow from CFD to ES indicates the transfer of $\Delta T_{i,air}$ and $h_{i,c}$ while the arrow from ES to CFD indicates the transfer of T_i and $Q_{heat\_extraction}$)

Staged Coupling	Illustration of Methodologies
Static Coupling	*One Step*: ES → CFD CFD → ES
	Two Step: ES → CFD → ES CFD → ES → CFD
Dynamic Coupling	*One-Time-Step Dynamic Coupling*: ES ⇄ CFD [Iterate till convergence at a specific time step]
	Quasi-Dynamic Coupling: ES → CFD → ES → CFD → 1st step, 2nd step [A period of time]
	Full Dynamic Coupling: ES ⇄ CFD → ES ⇄ CFD → 1st step Iterate till convergence; 2nd step Iterate till convergence [A period of time]
	Virtual Dynamic Coupling: ES ⇄ CFD [Different situations] ↓ Generate Functions DataBase of $\Delta T_{i,air}$ and $h_{i,c}$ ↓ ES [Year round]

Iteration of ES and CFD may result in convergence and stability problems due to the physical and numerical differences between ES and CFD programs. Different data-exchange methods in iteration may produce differences in convergence and stability behaviors. More theoretical and practical analysis of this topic may be expected in future papers.

In general, the building characteristics and the purpose of the simulation determine which coupling process is most suitable for a particular case. Several coupling processes may be used together to achieve the best solution for a specific case. For example, virtual dynamic coupling may be best for a whole year energy analysis, and one-time-step dynamic coupling may be adequate for equipment sizing.

CASE STUDY

The coupling strategies described above have been implemented using the EnergyPlus and MIT-CFD programs. EnergyPlus, developed for the U.S. Department of Energy, is a new energy simulation tool, based on DOE-2 and BLAST, that uses the Heat Balance Method described above in the paper. Developed at Massachusetts Institute of Technology, MIT-CFD is a general CFD tool, which can solve steady and unsteady laminar and turbulent flow problems with arbitrary geometry. Standard numerical methods and turbulence models are employed in MIT-CFD. A prototype version of the coupled EnergyPlus/MIT-CFD codes has been produced and used in the following preliminary case study.

The case study uses an office room to demonstrate a typical coupling calculation – a *quasi-dynamic coupling calculation* -- for a winter design day. The office is on a middle floor of a building located in Boston. It has only one exterior wall (the x-z plane in Figure 3), which faces south. The parameters of enclosure materials are listed in Table 2. There are no internal heat gains in the office, and the heating load is solely due to the south exterior wall. The room is conditioned 24 hours a day with a VAV system, as shown in Figure 3. The air-exhaust duct is precisely above the air-supply duct on the west wall. The supply air temperature is fixed at 30ºC, while the room temperature is controlled at 16ºC.

In this case, the CFD calculation is called every hour by ES for a period of four design days. Within the quasi-dynamic coupling process, ES first produces a set of surface temperatures and a heating load at the first hour and passes them to CFD. Based on these boundary conditions, CFD calculates the flow and temperature patterns for the first hour. Then ES obtains the $\Delta T_{i,air}$ and $h_{i,c}$ from the CFD results for the second run at the second hour, and so on. For simplicity, the ES assumption of isothermal surfaces is also adopted in the CFD.

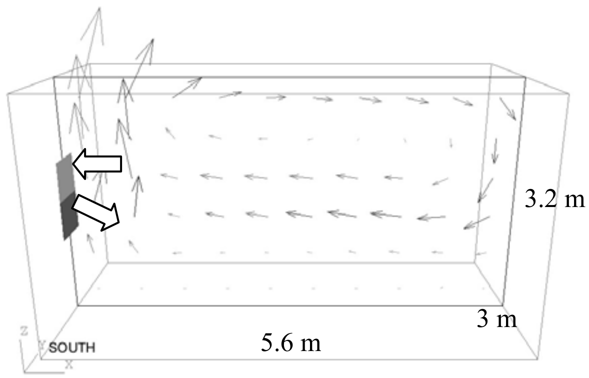

Figure 3. Configuration of the office and flow pattern

Table 2. Room enclosure materials

Enclosure	Thickness (m)	Density (kg/m^3)	Specific heat (J/kgK)	Thermal cond (W/mK)
Ceiling/ Floor	0.175	2300	840	1.9
Walls	0.140	700	840	0.23

The CFD program uses a zero-equation turbulence model (Chen and Xu 1998). The convergence criterion for the CFD is that the normalized residuals fall to less than 1% for all the variables solved. The total computing time for the coupled ES and CFD simulation is only 83 seconds on a PCIII-600 because the CFD solution uses an extremely coarse grid (10x5x6).

The results show that the heating load variation during the design day is not significant because of the weak solar effect in Boston's winter and the good insulation of the south wall. In the room, as seen in Figure 3, the low-velocity warm supply air comes into the space and goes up directly due to the strong buoyancy effect. The warm air flows back along the center-line, which forms a warm re-circulation region in the top part of the space. Since the temperature stratification in this case exists between the top and bottom levels of the room (about 3-4K), the average air temperature close to the south wall is almost as same as the controlled room air temperature that is represented in Table 3 by the small $\Delta T_{r,i}$. However, the interior convective heat transfer coefficient of the south wall calculated by MIT-CFD is almost twice as large as the one originally used in EnergyPlus. Hence, with this increased convective heat transfer coefficient, EnergyPlus predicts a greater heat flow from the room air to the surface, which also increases the surface temperature, as shown in Table 3.

Figure 4 presents the thermal performance of the south wall in terms of conduction, convection and radiation. The south wall gains heat from room air and other surfaces by convection and radiation, respectively, and then transfers the heat to the outside by conduction through the wall. The increased convective heat transfer in the coupled ES/CFD calculation increases the total heating load requirement by 9.4%. The heating load increase may be greater for the case with windows on the south wall due to the more important role convection plays in that case (Kendrick, 1993).

Table 3. Comparison of day-averaged values without and with CFD

South Wall	$h_{i,c}$ (W/m^2K)	$\Delta T_{r,i}$ (C)	T_{wall} (C)	Q (W)
Without CFD	2.41	0	9.62	583
With CFD	4.37	-0.1077	11.65	638

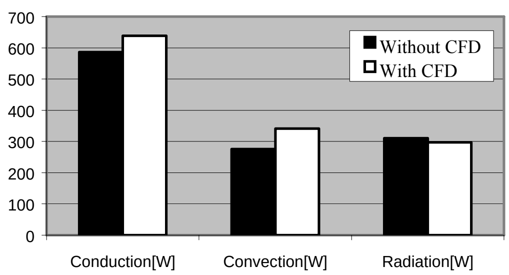

Figure 4. Thermal performance on south wall of the office

CONCLUSIONS

(1) It is beneficial to couple ES and CFD. Most significant thermal and flow assumptions used by individual ES and CFD programs can be eliminated by coupled simulation due to the complementary nature of the information provided by the programs.

(2) The conjugate heat transfer method and the code coupling method are two major methods of coupling ES and CFD. The former method may give better solutions, but the computational expensive makes it impractical within today's conditions. However, the latter method greatly reduces computing time by

using quasi-steady CFD simulation rather than unsteady CFD simulation, while providing acceptable solutions.

(3) In order to bridge the gap in computing speed between ES and CFD programs, a staged coupling strategy is proposed that may optimize the coupling process with a satisfactory solution as well as an acceptable computing effort. The building characteristics will be the main determinant of the most suitable coupling process.

This paper also presents a preliminary case study of an empty office room under winter design conditions in Boston, using an implemented coupling platform connecting EnergyPlus and MIT-CFD. The study illustrates a typical quasi-dynamic coupling process and indicates some performances of this platform. The results show an increased heating load requirement (about 10%) for this case with the coupled simulation due to the increased interior heat transfer coefficient from CFD calculation, which may be expected to be more significant in a case with windows on the wall.

ACKNOWLEDGEMENT

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Office of Building Technology, State and Community Programs of the U.S. Department of Energy under Contract No. DE-AC03-76SF00098.

REFERENCES

Beausoleil-Morrison I. (2000), ‘The Adaptive Coupling of Heat and Air Flow Modeling within Dynamic Whole-Building Simulation’, *PhD Thesis*, University of Strathclyde, Glasgow UK.

Chen Q. and van der Kooi J. (1988), ‘ACCURACY — A Computer Program for Combined Problems of Energy Analysis, Indoor Airflow, and Air Quality’, *ASHRAE Transaction*s, 94 (2) 196-214.

Chen Q., Peng X., and van Paassen A.H.C. (1995), ‘Prediction of Room Thermal Response by CFD Technique with Conjugate Heat Transfer and Radiation Models’, *ASHRAE Transaction*s, 3884 50-60.

Chen Q. and Xu W. (1998), ‘A Zero-Equation Turbulence Model for Indoor Airflow Simulation’, *Energy and Building*s, 28 137-144.

Clarke J.A., Dempster W.M., and Negrao C. (1995a), ‘The Implementation of a Computational Fluid Dynamics Algorithm within the ESP-r System’, *Proc. Building Simulation ’9*5, 166-75, Int. Building Performance Simulation Association, Madison USA.

Clarke J.A., Hensen J.L.M. and Negrao C.O.R. (1995b), ‘Predicting Indoor Air Flow by Combining Network Approach, CFD and Thermal Simulation’, *Proc. 16th AIVC Conf.*, 145-53.

Crawley, D. B., L. K. Lawrie, C. O. Pedersen, F. C. Winkelmann, 2000. “EnergyPlus: Energy Simulation Program”. *ASHRAE Journal*, Vol. 42, No. 4 (April), pp. 49-56.

Holmes M.J., Lam J.K.-W., Ruddick K.G., and Whittle G.E. (1990), ‘Computation of Conduction, Convection, and Radiation in the Perimeter Zone of an Office Space’, *Proc. ROOMVENT ’9*0, Oslo Norway.

Kendrick J F. (1993), 'An overview of combined modelling of heat transport and air movement', Technical Note 40, *Air Infiltration and Ventilation Centre*, Coventry, UK.

Moser A., Schalin A., Off F., Yuan X. (1995), ‘Numerical Modelling of Heat Transfer by Radiation and Convection in an Atrium with Thermal Inertia’, *ASHRAE Transaction*s, SD-95-14-4.

Negrao C.O.R. (1995), 'Conflation of Computational Fluid Dynamics and Building Thermal Simulation', *PhD Thesis*, University of Strathclyde, Glasgow UK.

Schild P. (1997), 'Accurate Prediction of Indoor Climate in Glazed Enclosures', *PhD Thesis*, Norwegian University of Science and Technology, Trondheim Norway.

Srebric, J., Chen, Q., and Glicksman, L.G. (1999). 'A Coupled Airflow-and-Energy Simulation Program for Indoor Thermal Environmental Studies', *ASHRAE Trans.* to be published.

Thompson, C. P., Leaf, G. K. (1988). 'Application of a Multigrid Method to a Buoyancy-Induced Flow Problem', *Multigrid Methods--Theory, Applications and Supercomputing*. S. F. McCormick. New York, arcel Dekker, Inc. 1: 605-629.